شبكة المعلومات الجامعية

بسم اللّه الرحمن الرحيم

شبكة المعلومات الجامعية

التوثيق الالكتروني والميكروفيلم

شبكة المعلومات الجامعية

جامعة عين شمس

التوثيق الإلكتروني والميكروفيلم

قسم

نقسم باللّه العظيم أن المادة التي تم توثيقها وتسجيلها علي هذه الأقراص المدمجة قد أعدت دون أية تغيرات

يجب أن

تحفظ هذه الأقراص المدمجة بعيدا عن الغبار

شبكة المعلومات الجامعية

بعض الوثائق الأصلية تالفة

بالرسالة صفحات
لم ترد بالأصل

CFD SIMULATION AND PARAMETRIC ANALYSIS OF AIRFLOW DISTRIBUTION AROUND CHILLERS IN A MECHANICALLY VENTILATED ROOM

By

Mohamed Rashad Mohamed Ali

A Thesis Submitted to the

Faculty of Engineering at Cairo University

In Partial Fulfillment of the Requirements for

The Degree of

Master of Science

In

MECHANICAL POWER ENGINEERING

FACULTY OF ENGINEERING, CAIRO UNIVERSITY
GIZA, EGYPT
2020

CFD SIMULATION AND PARAMETRIC ANALYSIS OF AIRFLOW DISTRIBUTION AROUND CHILLERS IN A MECHANICALLY VENTILATED ROOM

By

Mohamed Rashad Mohamed Ali

A Thesis Submitted to the

Faculty of Engineering at Cairo University

In Partial Fulfillment of the Requirements for

The Degree of

Master of Science

In

MECHANICAL POWER ENGINEERING

Under Supervision of

Prof. Dr. Essam E. Khalil Hassan Khalil

Professor
Mechanical Power Engineering Department
Faculty of Engineering, Cairo University

Dr. Muhammed Abdullah Hassan Ahmed

Assistant Professor
Mechanical Power Engineering Department
Faculty of Engineering, Cairo University

FACULTY OF ENGINEERING, CAIRO UNIVERSITY
GIZA, EGYPT
2020

CFD SIMULATION AND PARAMETRIC ANALYSIS OF AIRFLOW DISTRIBUTION AROUND CHILLERS IN A MECHANICALLY VENTILATED ROOM

By

Mohamed Rashad Mohamed Ali

A Thesis Submitted to the

Faculty of Engineering at Cairo University

In Partial Fulfillment of the Requirements for

The Degree of

Master of Science

In

MECHANICAL POWER ENGINEERING

Approved by the Examining Committee:

Prof. Dr. Essam E. Khalil Hassan Khalil (Thesis Main Advisor)

Professor, Mechanical power Engineering Department, Faculty of Engineering, Cairo University.

Prof. Dr. Mostafa Abdel-Hamid Rizk (Internal Examiner)

Professor, Mechanical power Engineering Department, Faculty of Engineering, Cairo University.

Prof. Dr. Ahmed Farouk Abdel-Gawad (External Examiner)

Professor, Mechanical power Engineering Department, Faculty of Engineering, Zagazig University

FACULTY OF ENGINEERING, CAIRO UNIVERSITY
GIZA, EGYPT
2020

Engineer: Mohamed Rashad Mohamed Ali
Date of Birth: 23/05/1994
Nationality: Egyptian
Email: immohamed_2010@yahoo.com
Phone: 01061193177
Address: El Maadi, Cairo, Egypt
Graded:
Registration Date: 01/03/2019
Awarding Date: … /.../ 2020
Degree: Master of Science
Department: Mechanical Power Engineering
Supervisors: Prof. Dr. Essam E. Khalil Hassan
Dr. Muhammed Abdullah Hassan
Examiners: Prof. Dr. Essam E. Khalil Hassan (Thesis Main Advisor)
Prof. Dr. Mostafa Abdel-Hamid Rizk (Internal Examiner)
Prof. Dr. Ahmed Farouk Abdel-Gawad (External Examiner)
(Professor, Mechanical power Engineering Department, Faculty of Engineering, Zagazig University)

Title of Thesis: **CFD simulation and parametric analysis of airflow distribution around chillers in a mechanically ventilated room.**

Keywords: Air cooled chiller, Indoor Package unit, Mechanical-room, airflow distribution, CFD simulation.

Summary:

For cases when it is difficult to introduce sufficient amounts of natural ventilation air to air-cooled chillers, mechanical ventilation is required. A typical example is mechanical/chiller rooms. Due to the confined space, part of the chillers' exhaust air is expected to be circulated back as part of the total intake air, eventually decreasing the coefficient of performance (COP) of the chillers. The objective of this study is to simulate the airflow distribution around chillers in such mechanical-rooms and to analyze the impact of room height, locations of the intake and exhaust louvers, number of chillers, and intake air velocity on the COP of chillers. This is achieved by developing a computational fluid dynamics (CFD) model in ANSYS® 19.0. The study shows that the locations of the intake and exhaust air-louvers and the intake louver air velocity are the most influential parameters that can significantly affect the performance of the chiller, and the lower this velocity is, the higher the chiller's intake air temperature, the best results were achieved at an intake air velocity of 2 m/s. The best configuration for the air-louvers is to introduce the air from two side inlets and to locate the exhaust louver at the top of the room. Finally, the room's clear height also affects the recirculation process, but it has a lower impact when compared to other parameters, the best results were achieved at a mechanical-room height of 6 m.

Disclaimer

I hereby declare that this thesis is my own original work and that no part of it has been submitted for a degree qualification at any other university or institute.

I further declare that I have appropriately acknowledged all sources used and have cited them in the references section.

Name: Mohamed Rashad Mohamed Ali Date:

Signature:

ACKNOWLEDGMENT

I would like to express my deep gratitude to **Prof. Dr. Essam E. Khalil** and **Dr. Mohammed Abdullah** for their advice, guidance, and continuous support. They provided me with valuable technical knowledge and information that were of a great support and help in my work.

Many thanks to **Dr. Ahmed El-Degwy** for his very useful help and support, especially in the modeling process.

I extend my gratitude to **Eng. Menna El-Shimi** for sharing her knowledge and experience and useful discussions.

Finally, I owe a lot of things to my parents, brothers, and friends for their encouragement, care, and patience.

ABSTRACT

For cases when it is difficult to introduce sufficient amounts of natural ventilation air to air-cooled chillers, mechanical ventilation is required. A typical example is mechanical/chiller rooms. Due to the confined space, part of the chillers' exhaust air is expected to be circulated back as part of the total intake air, eventually decreasing the coefficient of performance (COP) of the chillers. The objective of this study is to simulate the airflow distribution around chillers in such mechanical-rooms and to analyze the impact of room height, locations of the intake and exhaust louvers, number of chillers, and intake air velocity on the COP of chillers. This is achieved by developing a computational fluid dynamics (CFD) model in ANSYS® 19.0. The study shows that the locations of the intake and exhaust air-louvers and the intake louver air velocity are the most influential parameters that can significantly affect the performance of the chiller, and the lower this velocity is, the higher the chiller's intake air temperature. The best results were achieved at an intake air velocity of 2 m/s. The best configuration for the air-louvers is to introduce the air from two side inlets and to locate the exhaust louver at the top of the room. Finally, the room's clear height also affects the recirculation process, but it has a lower impact when compared to other parameters. The best results were achieved at a mechanical-room height of 6 m.

Keywords: *Air-cooled chiller; Indoor package unit, Mechanical-room, Airflow distribution; CFD simulation.*

TABLE OF CONTENTS

LIST OF TABLES